XVI International Conference on Computational Plasticity. Fundamentals and Applications
COMPLAS 2021
E. Oñate, D. Peric, M. Chiumenti and Eduardo de Souza Neto (Eds)

DIGITAL TWINNING TO PREDICT THE RESIDUAL LIFE OF COMPOSITE PRESSURE VESSELS – COMPLAS 2021

Nesrine Klebi[1*], Pierre Kerfriden[1,2], Basile Marchand[1], Alain Thionnet[1]

[1] Mines ParisTech, PSL University, MAT - Centre des Matriaux, CNRS UMR 7633, BP 87, 91003 Evry, France
nesrine.klebi@mines-paristech.fr, pierre.kerfriden@mines-paristech.fr

[2] Cardiff University, School of Engineering, Queen's Buildings, The Parade, Cardiff, CF24 3AA, United Kingdom

Key words: CPV (Composite Pressure Vessels), Numerical homogenization, FE^2, Composites, Micro-mechanics, Representative Volume Element (RVE)

Abstract. This paper presents an homogenization strategy to determine a deterministic constitutive model for composite damage quantification, based on a simplified stochastic FE^2 model and the validation of this strategy using a numerical test.

1 INTRODUCTION

The simulation of an in-service composite structure failure remains a major challenge in engineering. This is mainly due to the non-homogeneous nature of these materials. In particular, the damage evolution mechanisms, as they appear on a relatively small scale compared to the scale of the structure, plus the brittle aspect of the failure. Current solutions to diagnose damage patterns in a composite structure consist in carrying out Non Destructive Testing [1], using either active or passive NDT, such as acoustic emission-based systems. However, when employing these techniques, one is not able to describe neither the overall state of the structure and its degradation, nor its remaining lifetime. To overcome these difficulties, we propose a damage prediction approach via the use of digital twinning, where a numerical model will continuously assimilate data from accoustic emission sensors and provide damage states as time advances.

Two scale Finite Element models, such as FE^2 , can in principle be utilized to predict the evolution of damage within the microstructure of composite materials. However, due to its extreme computational cost, many alternative-FE^2-based techniques are put forward to take advantage of the expressive power of material-point-level FEM models whilst ensuring computationally tractable maroscale simulations. One known approach is the process of meta-modeling [2, 3], where one construct an homogeneous constitutive model at the mesoscale that implicitly embeds the FE^2 constitutive generality and simplifies it.

In this work, we will present an homogenization strategy to determine a deterministic mesoscopic constitutive model based on an improved FE^2 model [4, 5] and the validation of this model on a numerical test.

2 THE MULTISCALE FIBRE BREAK MODEL DEVELOPED AT MINES PARISTECH

A stochastic mesoscopic model described also as a simplified FE^2 model, was developed at Mines Paristech first by Blassiau [4] and then was used by Thionnet [5] to quantifie damage process leading up to failure. This model is based on a ***two* scale approach** that provides detailed simulations, considering many physical phenomena,which were identified experimentally on specimens using acoustic emission control [7, 8] and high resolution tomography.

The first scale of the model is the *microscale*, where the composite is seen to be formed with the epoxy matrix surrounding carbon fibers. For a fiber volume fraction $V_f \approx 0.64$, the size of the Representative Volume Element has been deduced from the work of Baxevanakis [9], who was interested in micromechanics and tests at fibers scale. In addition, multi-fragmentation tests were performed on a single fiber embedded in the matrix in order to evaluate the length of the weakest link of a fiber ($\approx 0.5mm$), and by varying the number and the lengths of fibers in a bundle, it was numerically found that beyond the use of 6 fibers and a length $L = 4mm$, the strength of the composite material converged. Thus the conducted numerical study allowed to define the size of the 2D-RVE, besides it also showed that fiber breaks were concentrated in the same plane where each fiber only breaks once when its lentgth is $\approx 0.5mm$. This latter characteristic added to some geometrical constraints allowed Blassiau [7] to extend the 2D-RVE into a 3D-RVE consisting of 32 fibers arranged in a hexagonal array as shown in Figure 1.a.

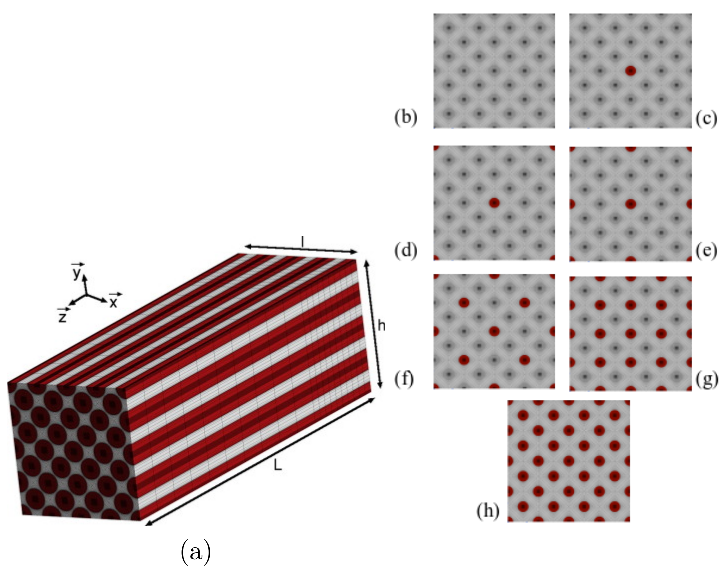


Figure 1: (a) Size of the cell representative of the RVE : L=4 mm, l=h=0.05 mm. (b),(c),(d),(e),(f),(g),(h) Representative cells of the material damage state and corresponding broken fibers are in red.[5]

The evolution of the number of fiber breaks during the lifetime of a structure requires in general the knowledge of the fibers strength properties. In addition, tensile tests were conducted on single fibers using different gauge length (25, 50, 100, 250 mm) and then extrapolated to a gauge length equal to the length of the already determined cell ($L = 4mm$), in order to experimentally calibrate a Weibull distribution of fiber strength. This made the corresponding 3D-RVE stochastic, where 6 representative cells of material damage states were considered : starting from a virgin state (Figure 1.b) to the completely damaged state (Figure 1.h). These states actually describes the evolution of the number of fiber breaks in the 3D-RVE when it is subject to a certain load.

The second scale of the model is the *macroscale*, where structures, namely composite pressure vessels, are modelled describing the overall behavior, geometry, mesh, and boundary conditions. In addition, a two-scale simplified FE^2 approach was developed to relate the previously described microscopic damage information to the macroscopic calculations. The latter improved FE^2 technique actually consists on homogenizing the behavior of the fibers and the matrix that forms the RVE into a single stochastic damageable orthotropic material. This homogenized constitutive law is then integrated locally at each Gauss point of the mesh while taking into account the fiber strengths designated by a MonteCarlo process that is associated with the Weibull distribution. Concerning the type of the mesh, a convergence study conducted by Blassiau [7] (using the size of the 3D-RVE) indicated that a hexahedral Finite Element that contains 8 Gauss points, where each embeds the model of the 3D-RVE, gave similar macroscopic stress field as one Finite Element containing 1 3D-RVE. Thus the size of 3D finite element (a x b x c) that must be used for analysing the composite structure is displayed in Figure 2 below.

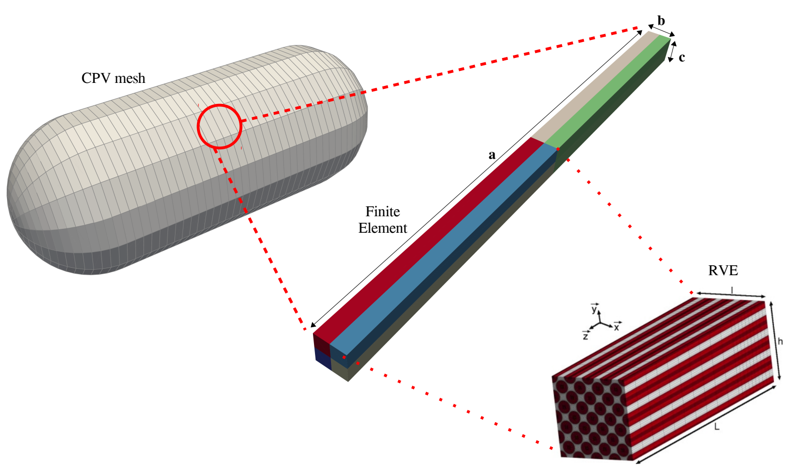


Figure 2: Characteristics of the simplified FE^2 calculation, 3D-RVE at the microscopic scale, 1 finite element c3d8 type, 8 nodes, 8 Gauss points, a=8 mm, b=c=0.1 mm [5]

The two-scale model detailed above [7] relates the microscale damage information to the macroscale calculations by updating the rigidity matrix at each time step of the finite element analysis using the equation below :

$$C_{11} = {C_{11}}^0 (1 - \frac{1}{NFC}) \tag{1}$$

where C_{11} is the updated stiffness in the fiber direction, ${C_{11}}^0$ is the current stiffness of the material (also in fiber direction), ad NFC are the number of fibres that are still undamaged. In addition to this, the modelisation of the Composite Pressure Vessel with this model will have an expensive computational cost. The cost is also due to the variability (stochasticity) that is described at a very small scale (size of the 3D-RVE) and to the invariant mesh which is refined even in unnecessary parts of the structure. Hence, the latter illustration lead us to the main objective, detailed in Section 3, where the stochastic part of this model [7] will be homogenized to a deterministic stress-strain law that represent the behavior before the failure (instability point) of the whole composite.

3 HOMOGENIZTION TECHNIQUE TOWARDS A DETERMINISTIC MESOSCOPIC DAMAGE MODEL FOR COMPOSITE PRESSURE VESSELS

The Composite Pressure Vessel, as shown in Figure 3, is made by filament windings process on both cylindrical and spherical parts that consecutively presents the cylinder and the dome. The composite material is then characterized by the stacking sequence of filaments and their thicknesses, as well as the material of the inner layer which is useful for separating the pressurized gas inside the tank from the composite material (the liner : metallic or polymer material).

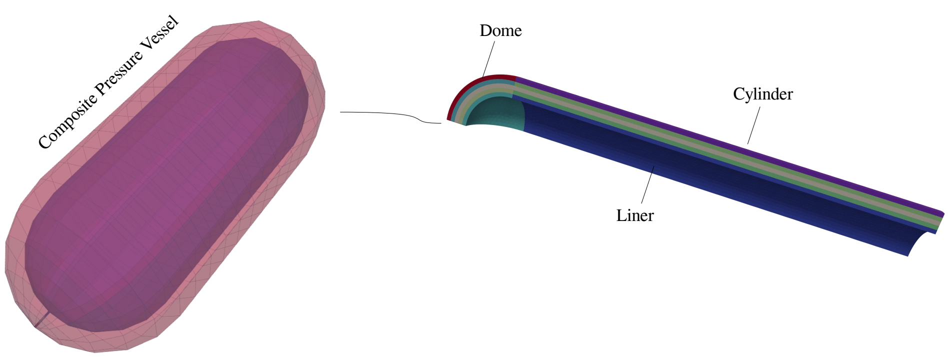


Figure 3: Composite Pressure Vessel components

The main idea, related to the end of Section 2, is to determine the homogenized mesoscopic constitutive law of the CPV using the existing simplified FE^2 numerical model previously mentioned. It is then assumed to realize tensile tests on specimens of different sizes using the same stacking sequence of the CPV. The searched homogenized model will give the possibility to overcome the constraint of the invariant mesh and will make it possible to mesh the structure with tetrahedron finite elements and to re-mesh the damaged layers in only one element at the scale of a ply, in order to reduce the computational cost. PS : The materials of the CPV are either orthotropic materials or damaged orthotropic materials (for the damaged ply) for composites and an elastic isotropic material for the liner.

The homogenization method can then be summed in 3 main steps presented by the following organizal chart.

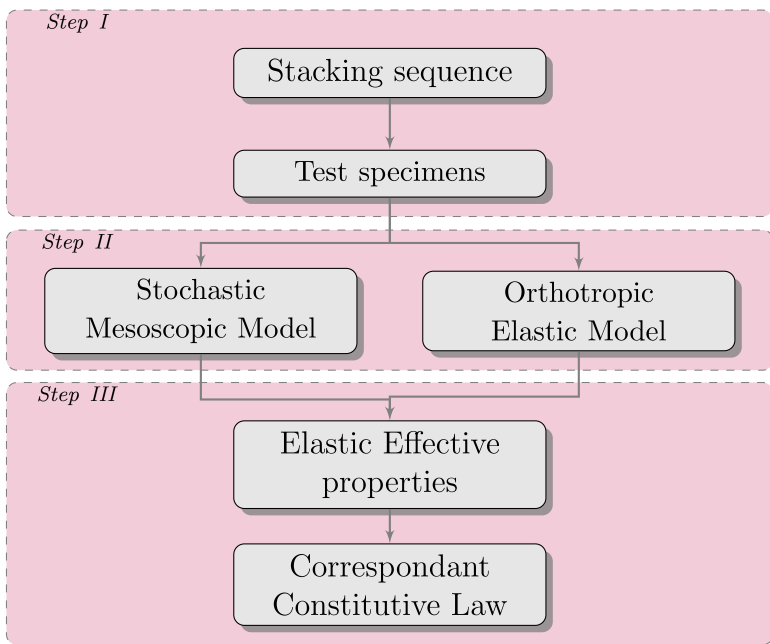


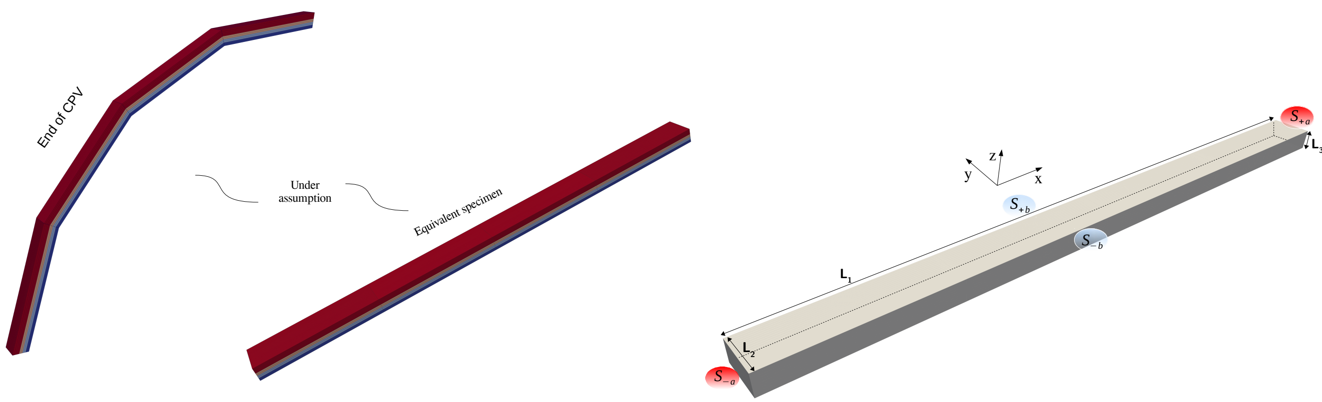
Figure 4: Organizal chart for explaining numerical homogenization steps

3.1 Detailed description of the homogenization strategy means

The first step, as seen in Figure 4, is to choose the type of the composite pressure vessel material on which will be performed the homogenization. It is important to mention that there exists no general rule for this step, therefore composite layer thicknesses choice is highly depending on the application. In the studied case, the selected CPV represents the following charecteristics:

- The stacking sequence [liner/-70°/+70°/90°]
- The damaged model is only applied in the upper layer (also called hoop layer)

The second step, as depicted in Figure 4, is considering that infinite radius of curvature reservoir are simplified to a parallelipepid specimen of the same material, the tensile tests will be carried out only on the equivalent parallelipipedic specimens described on Figure 5.a



(a) (b)

Figure 5: (a) 3D view of a CPV End and Equivalent test specimen, (b) Geometrical details of the Equivalent test specimen

The third step, illustrated in Figure 4, consists on performing 10 MonteCarlo simulations (to be statistically representative) using the existing model [7] for each specimen size detailed in Table 1.

Specimen	L1 (mm)	L2 (mm)	L3 (mm)
1	32	4	1
2	64	8	1
3	128	16	1
4	256	32	1

Table 1: Dimension of the studied specimens

The numerical measurement process is also set up in this step, where two methods depending on the Boundary Conditions were applied to the test specimens in Figure 5.(b). The first method is the Neumann method, where Dirichlet loads $U_{+a}(M) = +U_{x_1}$ and $U_{-a}(M) = -U_{x_1}$ are applied respectively on S_{+a}, S_{-a}, while all other surfaces are left free of stress. The second method is the Dirichlet method where lateral diplacements are added to the boundary conditions of the Neumann method such that on S_{+b} and S_{-b}, are applied respectively $U_{+b}(M) = \frac{2x}{L} U_{x_1}$ and $U_{-b}(M) = \frac{2x}{L} U_{x_1}$.

The properties to be identified are the effective longitudinal stress-strain curve σ_{11} (longitudinal stress) according to ϵ_{11} (longitudinal strain) that is supposed to be the same on all the test-specimen, and the curve of τ_f (Number of Fibre Breaks per unit volume) according to ϵ_{11} (longitudinal strain). The latter curve will be useful for the post-processing of the results and for future work in data assimilation.

The direct justification for the use of both numerical measurement processes is for the sake of the homogenization theory. Where for sufficiently large test specimen, the mean curves σ_{11} - ϵ_{11} and τ_f - ϵ_{11} with no lateral displacement coincide with the same curves when adding the lateral displacements due to the elimination of the effect of the geometry of the structure and the boundary conditions.

3.2 Homogenization step

The failure information that occurs, in the microscopic model, at the scale of the stochastic Representative Volume Elementary **(RVE)** composed by the matrix and the 32 fibers is related to the fracture information at the macroscopic scale of the structure as already explained in Section 2. This relationship is governed by the updating of the first component of elasticity tensor at each finite element calculation time-step . The material used in the existing model [7] is an orthotropic damaged material, whose constitutive law is of the following form :

$$\begin{pmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \sigma_4 \\ \sigma_5 \\ \sigma_6 \end{pmatrix} = \begin{pmatrix} C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\ C_{12} & C_{22} & C_{23} & 0 & 0 & 0 \\ C_{13} & C_{23} & C_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & C_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & C_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & C_{66} \end{pmatrix} \begin{pmatrix} \epsilon_1 \\ \epsilon_2 \\ \epsilon_3 \\ \epsilon_4 \\ \epsilon_5 \\ \epsilon_6 \end{pmatrix} \tag{2}$$

Given that in component C_{11} only the stiffness modulus is variable :

$$C_{11} = \frac{(1 - \mu_{23}\mu_{32})E_1}{\delta} \tag{3}$$

$$\delta = 1 - \nu_{12}\nu_{23}\nu_{31} - \nu_{13}\nu_{21}\nu_{32} - \nu_{12}\nu_{21} - \nu_{13}\nu_{31} - \nu_{23}\nu_{32}. \tag{4}$$

To be able to find the value of longitudinal stiffness modulus E_1, the call for the tensor of flexibility is mandatory, whose first component is :

$$S_{11} = \frac{1}{E_1} \tag{5}$$

From this formula one will determine the value of E_1 and will be able to express C_{11} as function of the longitudinal stiffness modulus for each finite element calculation time step.

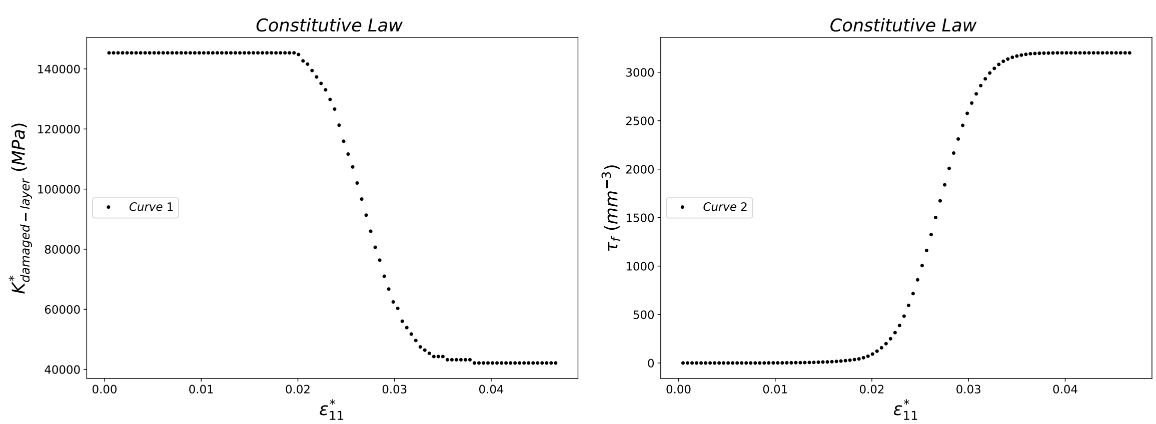
The fourth step, as illustrated in Figure 4, is then to perform an elastic orthotropic calculation on the largest specimen. This specimen should be large enough so the effect of the geometry and boundary conditions are eliminated (Section 3.1), and should be remeshed with very fine tetrahedral mesh since the objective is to characterize a continuous model. The elastic calculation should then achieved while varying the longitudinal stiffness modulus E_1 for the damaged layer at each time step of the simulation.

The final step, in Figure 4, is to post-process the results of step 3 and step 4. Hence, providing the average responses of the 10 Montecarlo simulations, and the equivalent elastic response, interopolaton of both results will yield to the homogenized constitutive law.

4 NUMERICAL RESULTS

4.1 The deterministic mesoscopic model determined by the homogenization strategy

The constitutive law can finally be defined and summarized in two curves which are respectively the longitudinal stiffness modulus of the damaged layer and the Number of Fibre breaks per unit volume τ_f as a function of the longitudinal deformation as precised in Figure 6.

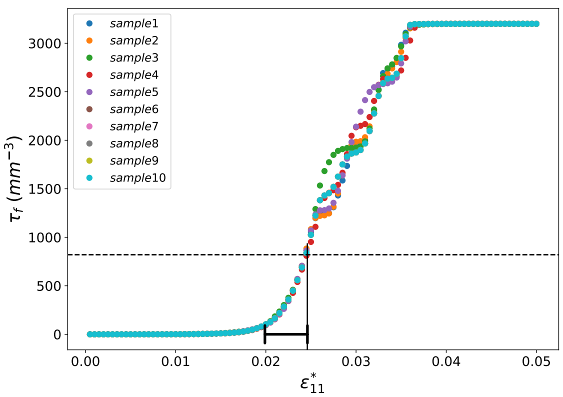


(a) $K^*_{damaged-layer}$- ϵ^*_{11} (b) τ_f - ϵ^*_{11}

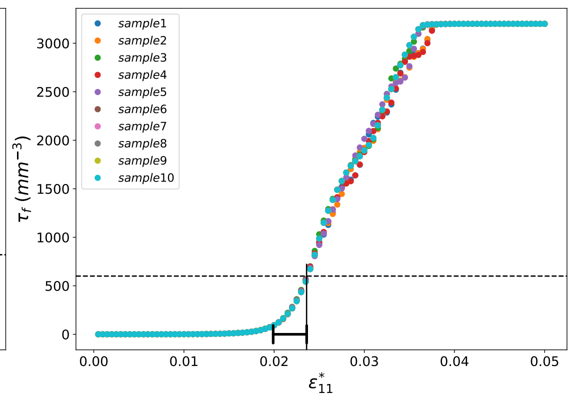
Figure 6: The homogenized constitutive behavior

4.2 MonteCarlo results and characterization of the instability point

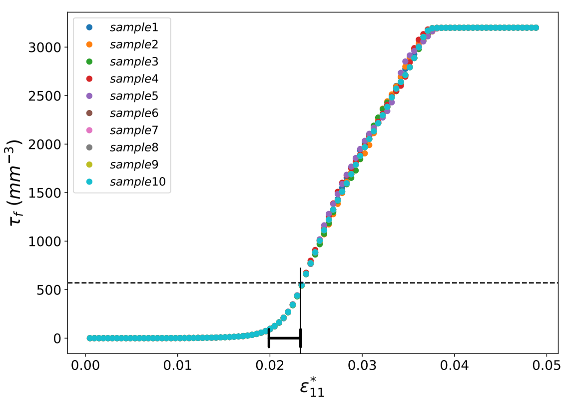
Figures 7 shows below the 10 Montecarlo simulations responses of the Number of fiber Breaks per unit volume τ_f as a function of the effective longitudinal deformation ϵ_{11}^* of the 4 specimens illustrated in Table 1.



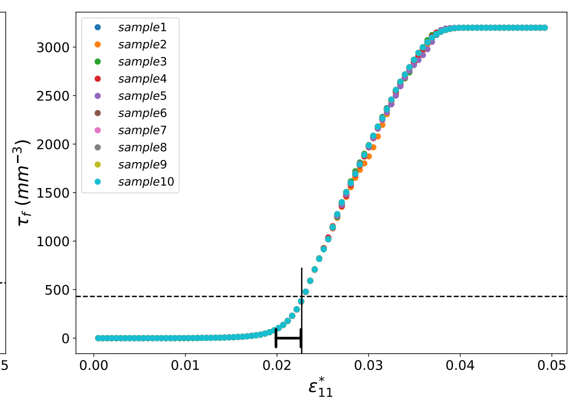
(a) Specimen 1



(b) Specimen 2



(c) Specimen 3



(d) Specimen 4

Figure 7: tensile tests response curves τ_f - ϵ_{11}^* of the tested specimens

The Analysis of the above sub-figures can lead to a fact that the composite failure point (also called the instability point) is reached more quickly as the volume of the damaged layer increases. One can see this latter as the point where the MonteCarlo runs start to disperse and the damage localization bands begin to appear in the specimen. As a result this dispersion could be quantified and the failure point could be characterized by assigning a percentage of this dispersion to it for each studied size. Then, a Weibull model could be fitted for the data as demonstrated by Figure 8 (for prediction of failure).

$$P(V,\sigma) = 1 - e^{-V(\frac{\sigma-\sigma_u}{\sigma_0})^m} \tag{6}$$

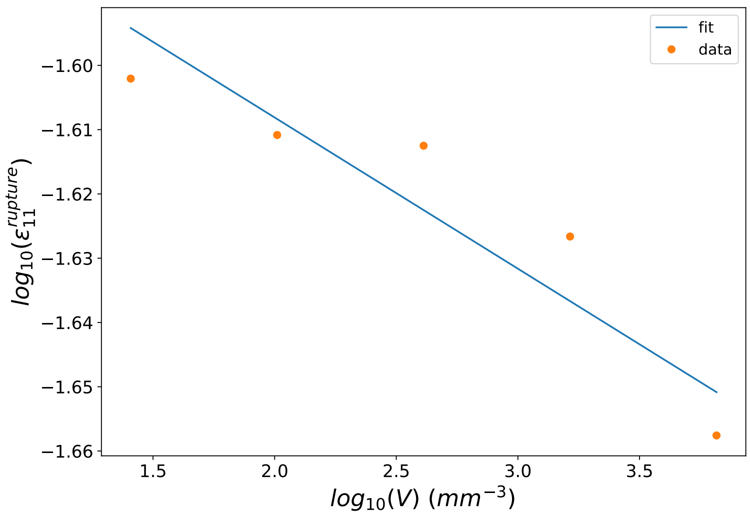


Figure 8: Weibull Model fitted to the numerical failure points

4.3 Performance validation of the deterministic mesoscopic model

Two calculations were realized using the existing model with the invariant hexahedral mesh and using the deterministic model on a tetrahedral mesh as shown in Figure 9.

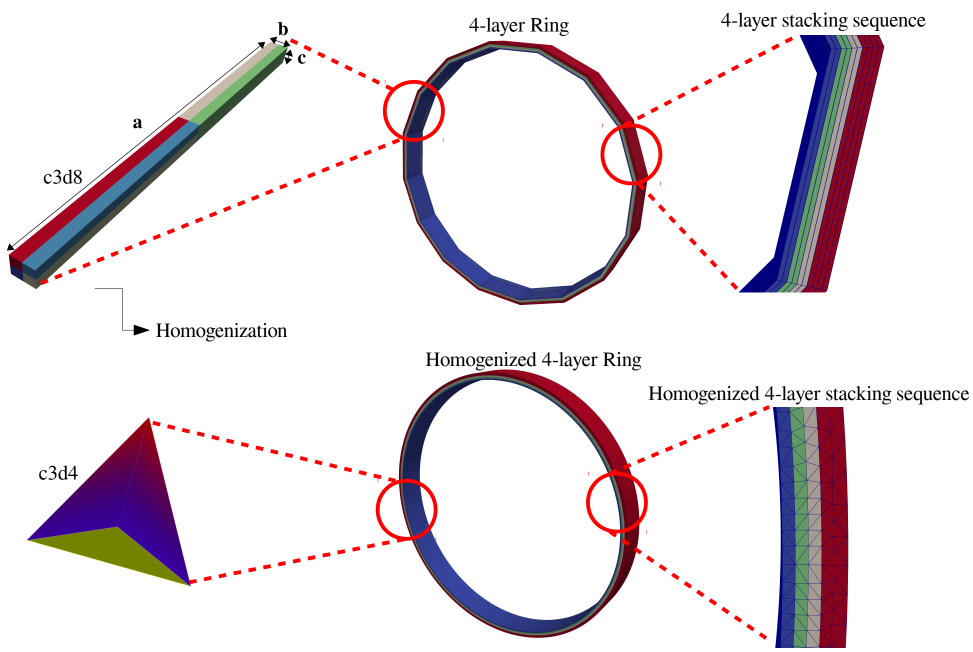


Figure 9: Details of the homogenization strategy modelisation on a Ring

The results of both calcultions before the instability point (failure point) are shown in Figure 10.

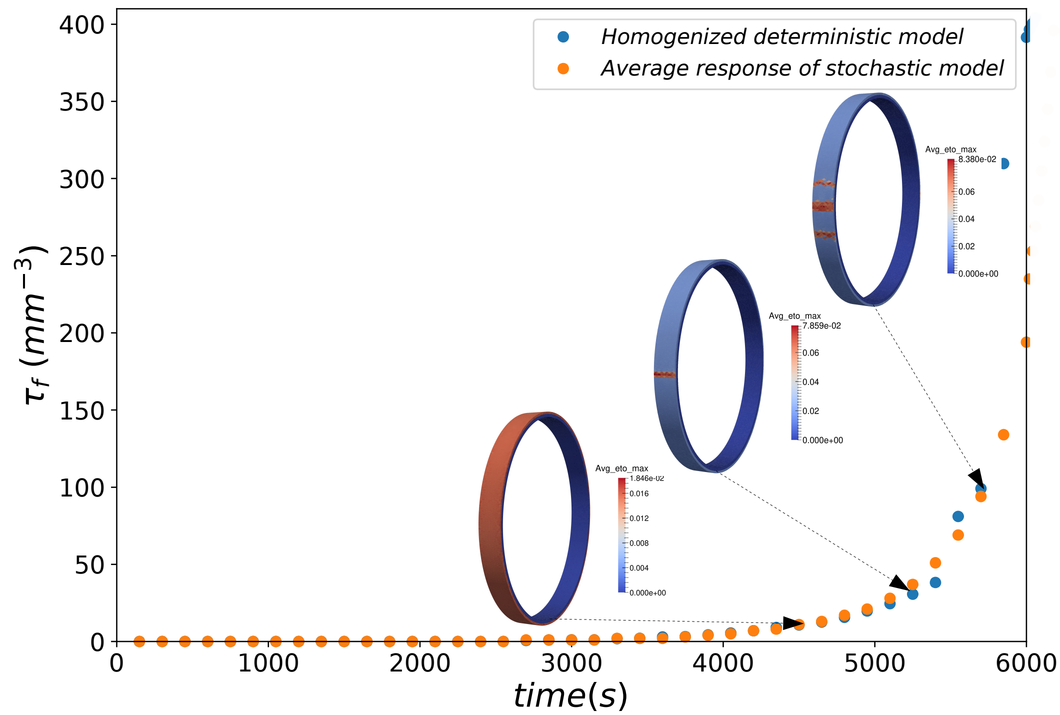


Figure 10: A performance test of the homogenized constitutive behavior on a Ring

5 CONCLUSION AND PERSPECTIVE

The homogenization strategy has been successfully performed and the deterministic mesoscopic model was determined. This behavior law was also implemented in the Finite Element Analysis software of Mines ParisTech (Zset). The performance of the new model was also tested, where two calculations performed using the stochastic model [7] and the deterministic model has lead to the same result of damage propagation before that the localisation occurs, consequently, the modelisation of the composite pressure vessel using the homogenized behavior is no more depending on the fine invariant hexahedral mesh. For future work of the dissertation, the deterministic model will be utilized to predict the residual lifetime of the CPV using data assimilation algorithms. Ensemble Kalman filter [6, 10] will be used for the estimation of the CPV state. In this case, the determined model will be merged with external data, namely acoustic hit frequencies measured by integrated sensors, and will manage to assimilate it and to construct sequential damage states with an optimal computational cost.

REFERENCES

[1] Mccrory J.P, Al-jumaili, S., Crivelli, D., Pearson, M., Eaton, M., Featherston, C. Guagliano, M., Holford, K., Pullin, R. *Damage classification in carbon fibre compos-*

ites using acoustic emission : A comparison of three techniques. Composites Part B: Engineering, 2015, 68. pp. 424–430.

[2] Rocha, I.B.C.M., Kerfriden., P., van der Meer, F.P. *On-the-fly construction of surrogate constitutive models for concurrent multiscale mechanical analysis through probabilistic machine learning*. Journal of Computational Physics: X, Volume 9, 2021, 100083

[3] Rocha, I.B.C.M., Kerfriden., P., van der Meer, F.P. *Micromechanics-based surrogate models for the response of composites: A critical comparison between a classical mesoscale constitutive model, hyper-reduction and neural networks*. European Journal of Mechanics - A/Solids, Volume 82, 2020, 103995

[4] Blassiau, S., Thionnet, A., R.Bunsell, A. *Micromechanisms of load transfer in a unidirectional carbon fibre-reinforced epoxy composite due to fibre failures: Part 3. Multiscale reconstruction of composite behaviour*. Composite Structures, Elsevier, 2008, 83. pp. 312-323.

[5] Thionnet, A., Chou, H.Y., R.Bunsell, A. *Fibre break processes in unidirectional composites. Composites Part A: Applied Science and Manufacturing*, Elsevier, 2014, 65. pp. 148-160.

[6] Kuzin, D., Yang, L., Isupova, O., Mihaylova, L. *Ensemble Kalman Filtering for Online Gaussian Process Regression and Learning*. 21st International Conference on Information Fusion (FUSION), 2018, pp. 39-46.

[7] Blassiau, S. *Modelisation des phénomènes microstructuraux au sein d'un composite unidirectionnel carbone/epoxy et prédiction de durée de vie: contrôle et qualification de réservoirs bobinés*. Thèse, Ecole des Mines de Paris, 2005.

[8] Chou, H., Zejli, H., Thionnet, A., Bunsell, A., Mouritz, A., Bannister, M. *Détection et discrimination par émission acoustique des endommagements dans les composites stratifiés d'unidirectionnels: rupture de fibre, macro et microfissuration intralaminaire, microdélaminage. comparaison expérience/ modélisation*. Matériaux, 2010, pp. 18–22 octobre, Nantes, France, 2010

[9] Baxevanakis, C. *Comportement statistique a rupture des composites stratifiés*. Thése, Ecole des Mines de Paris, 1994.

[10] Marchand, B. *Assimilation de données et recalage rapide de modèles mécaniques complexes*. Thése, Ecole Normale Supérieure Paris-Saclay , 2017.